Journal of Visualization, Vol. 7, No. 3 (2004) 249-256

Visualization of Secondary Flow Choking Phenomena in a Supersonic Air Ejector

Desevaux, P.*, Mellal, A.* and Alves de Sousa, Y.*

* CREST / CNRS - UMR 6000, University of Franche-Comté, IGE, Parc Technologique, F-90000 Belfort, France.
E-mail: philippe.desevaux@univ-fcomte.fr

Received 3 December 2003
Revised 5 March 2004

Abstract : The present study deals with the visualization of the air flow inside a supersonic ejector. Our attention is more precisely focused on the choked flow phenomenon which occurs along the mixing chamber of the secondary nozzle and which can be visualized by CFD. Laser tomography visualizations are used to validate the CFD model. The evolution of flow configuration in the ejector with the primary stagnation pressure is examined both in the case of zero secondary flow and in the case of free entrainment of induced air.

Keywords : Ejector, Supersonic Flow, Shocks, Numerical Simulation, Laser Tomography.

1. Introduction

Ejectors are devices which are used in a wide range of applications (Sun and Eames, 1995): vacuum pumps, ejector-compression heat pumps, thrust augmenting systems, gas vapour recovery … In the majority of practical applications, ejectors are made of two coaxial nozzles. A high-pressure fluid (the driving or primary fluid) enters the primary nozzle, where it expands to produce a high-velocity jet. This entrains a low-pressure fluid (the secondary or induced fluid) at its boundary, and the two fluids are then combined in the mixing chamber (constant-area duct) of the secondary nozzle. The complex interaction between the two streams, the shock waves and the boundary layer, causes a very complicated flow in the secondary nozzle of the ejector (Matsuo et al., 1999, Chen et al., 1994). Many researches have been devoted to the analysis of the flow structure inside supersonic ejectors and some authors have proposed, thanks to flow visualization, a classification of the flow regimes (Fabri and Siestrunck, 1958, Matsuo et al., 1981, Matsuo et al., 1982). Among these flow regimes, we mainly distinguish between flow regimes with and without choking of the secondary nozzle. The transition between these two flow regimes, which essentially depends on stagnation pressure conditions and ejector geometry, is of great importance since it strongly affects the ejector performance in terms of suction entrainment capacities (Eames et al., 1995, Chou et al., 2001). Then, we better understand the interest to visualize the flow inside the secondary nozzle of the ejector, either by experimental techniques (Desevaux et al., 1995) or by numerical tools (Annamalai et al., 1998, Sankaran et al., 2002), to determine the flow regime in the ejector.

The object of this work is to investigate the capabilities of a CFD model to correctly visualize the flow inside a supersonic air ejector and to precisely predict the start of the flow regime with choking of the secondary nozzle. The CFD model developed by the present authors has already been applied to supersonic ejectors operating without choking of the secondary nozzle (Desevaux and

Aeschbacher, 2002, Desevaux et al., 2002). The originality of the present work is to extend the modelling of the air flow to more complex flow configurations with transition between flow regimes with and without choking of the secondary nozzle, and to validate the CFD model by comparing numerical flow visualizations with laser tomography images.

2. Experimental Apparatus

Figure 1 gives a schematic view of the ejector configuration and of the flow visualization system used in this work. High-pressure air (mass flow rate m_1, stagnation pressure P_1) enters the ejector to be accelerated to a supersonic velocity through the primary Laval nozzle. By an entrainment-induced effect, the secondary air (mass flow rate m_2, stagnation pressure P_2) is drawn into the flow and accelerated. Mixing and recompression of the resulting stream then occur in the secondary nozzle and especially along its constant-area mixing chamber. The primary-induced air mixture is finally discharged into the surrounding atmosphere at pressure Pa. The main dimensions of the two converging-diverging nozzles forming the ejector are shown in Fig. 1. Regarding this experimental setup, it is of interest to note that:

- The primary nozzle is expected to produce a supersonic flow with an exit Mach number $M_1 = 2.3$.
- We define the ejector entrainment ratio U as the ratio of the secondary mass flow rate to the primary one:

$$U = m_2/m_1$$

- The ejector has been designed with a throat-area ratio $(D/d^*)^2 = 4$.
- Compressed air is filtered before entering the primary nozzle to remove large particles (dust particles, compressor oil droplets) but is not dried. Therefore, the primary nozzle is fed with moist air.

Flow visualizations are carried out by laser sheet illumination methods. The optical arrangement presented in Fig. 1 uses a continuous wave argon laser, with a power of about 1.5 W in the blue line (λ = 488 nm) and emitting a vertically polarized laser beam. This laser beam is located at the ejector axis by means of mirrors. An oscillating mirror generates a dynamic light sheet (with a constant width slightly inferior to the mixing chamber diameter) reflected in the upstream direction along the ejector axis.

The marking of the flow is achieved by natural tracers. These natural tracers are water droplets issued from the condensation of moisture present in the primary and secondary fluids. A previous study of the light scattered by these droplets (Desevaux, 2001) has shown that they have a mean diameter less than 0.1 µm.

The viewing direction is perpendicular to the flow axis. The flow field area is divided into three

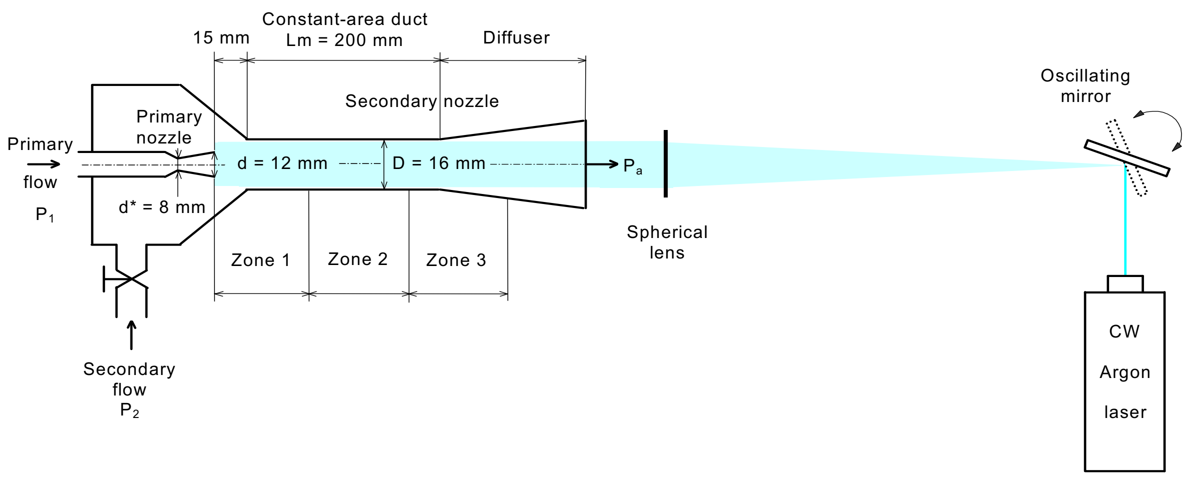

Fig. 1. Experimental setup.

zones of study as indicated in Fig. 1:
- zone 1 : immediate downstream region of the primary nozzle exit,
- zone 2 : middle region of the mixing tube,
- zone 3 : end of the mixing tube and entrance of the secondary diffuser.

3. Numerical Procedure

The commercial CFD package Fluent is used to model the flow within the ejector. The configuration of the ejector allows the 2D axisymmetric simulation of the flow. The computational grid, which is generated with the pre-processor Gambit, is a combination of structured mesh (for the primary nozzle and the mixing chamber which are both of simple geometry) and unstructured mesh (for the computational domain defined by the convergent part of the secondary nozzle). Converged, grid independent solutions were obtained for a refined grid of 15400 cells. The steady flow simulation is performed by using the coupled solver. Air is assumed to have the properties of an ideal gas.

Concerning the boundary conditions, a stagnation pressure is imposed at the inlet section of the primary nozzle, and the pressure at the outlet section of the ejector is fixed to the atmospheric pressure. Regarding the secondary flow inlet, we define an annular, coaxial section equivalent to the real test section. At this induced flow inlet, we impose a pressure inlet condition equal to the atmospheric pressure. For numerical simulations without entrainment of induced air, the secondary flow inlet is replaced by a wall boundary condition. The turbulence of the flow is modeled using the k-omega SST model, which has been found in a recent study (Bartosiewicz et al., 2003) to be well suited to predict supersonic flow with mixing and shock waves in ejectors.

4. Results and Discussion

4.1 Comparison between CFD and experimental flow visualizations

The first part of this study has consisted in comparing numerical flow visualizations obtained by CFD with laser tomography images of the flow inside the ejector. This comparison is realized for typical operating conditions of the ejector (i.e. $P_1 = 5.5$ bar, $U = 0.32$), for which the complex flow regime with choking of the secondary nozzle is supposed to appear (Matsuo et al., 1982). Figure 2 compares computational visualizations (Mach number and temperature fields, monochromatic images of Mach number field) with the laser tomography images of the flow in the three study zones. A few remarks can be made:
- In the zone 1, experimental and numerical flow visualizations are found in good agreement, especially for the location of the first shock and the shape of the overexpanded supersonic jet at the primary nozzle exit. Furthermore, the exit Mach number predicted by CFD (i.e. $M_1 = 2.34$) is in good accordance with the value of $M_1 = 2.3$ calculated from the quasi-1D theory for inviscid flows.
- In the zone 2, it may be noted that the Mach number decreases along the constant-area mixing chamber as the temperature increases. However, the flow remains supersonic along the entire length of this mixing tube and no shock wave is seen either in experimental or in CFD visualizations.
- In the zone 3, the flow passes to subsonic through a shock wave located at the beginning of the diffuser. This shock is observed experimentally as well as numerically, with however a slight shift in its location. We can also notice that the shock is not perfectly normal in the whole section of the diffuser, due to the viscous effect within the boundary layer along the diffuser wall.

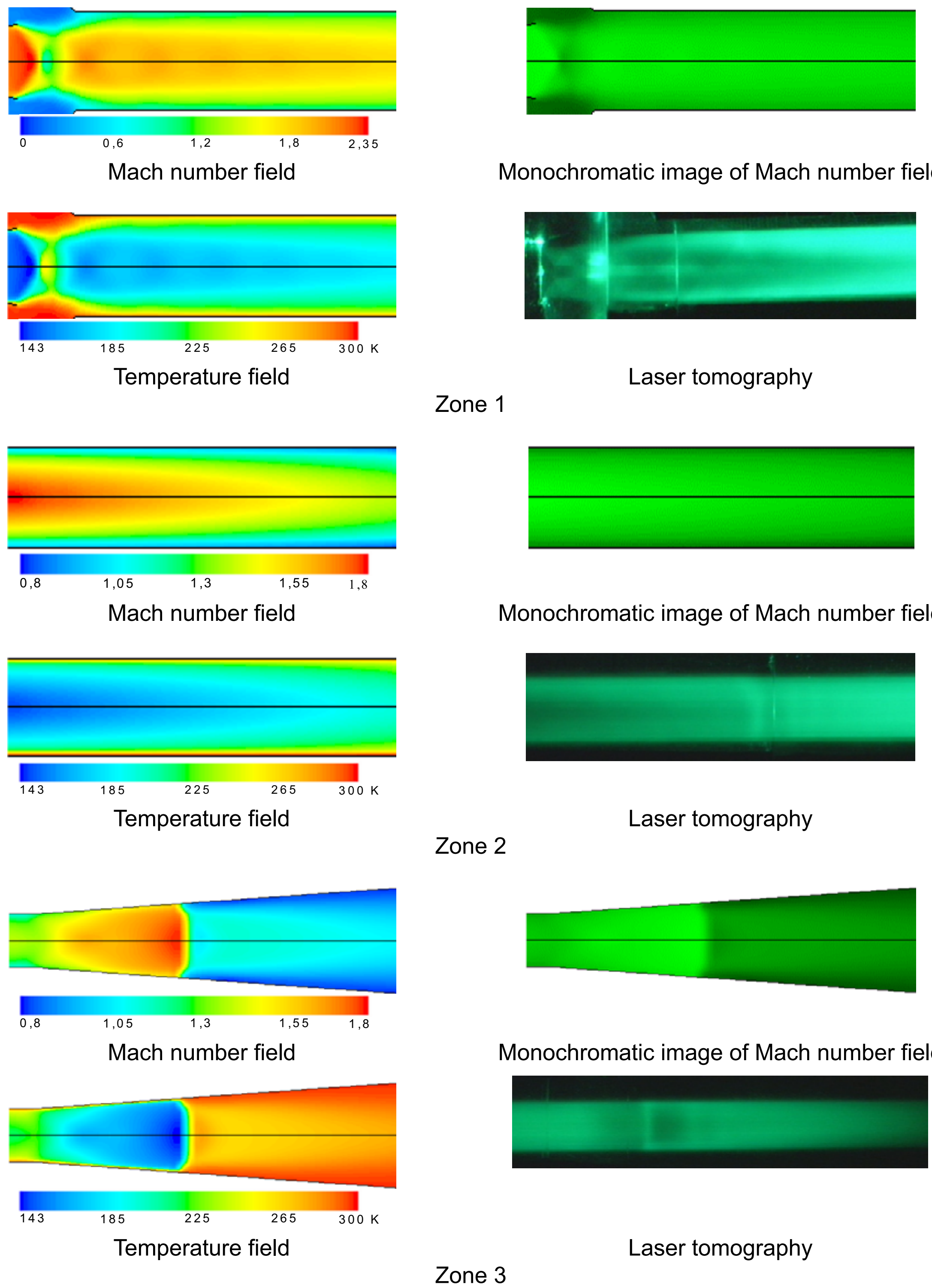

Fig. 2. Comparison between CFD and experimental flow visualizations within different regions of the secondary nozzle.

To summarize, we may conclude that the numerical flow visualizations obtained from the present CFD model are in fairly good agreement with the experimental ones and permit the successful prediction of the development of choked flow in the secondary nozzle.

4.2 Influence of the stagnation pressure

The second part of this study is devoted to the study of the evolution of the flow structure as a function of the primary stagnation pressure P_1. In this way, we present in Figs. 3 and 4 representations of the computed Mach number field inside the ejector (covering the distance from the middle section of the primary nozzle divergent to the middle section of the secondary diffuser). First results (Fig. 3) are related to the ejector operating without induced flow. By examining these pictures, it may be noticed that:

- For $P_1 < 3$ bar : The air flow passes from supersonic to subsonic through a quasi-normal shock in the primary nozzle divergent. In these conditions, the ejector operates in the subsonic regime.
- For $3 \leq P_1 \leq 3.5$ bar: The flow is entirely supersonic in the primary nozzle, continues to accelerate just downstream of the nozzle exit, and becomes subsonic through a series of oblique shocks in the mixing tube.
- For $3.5 < P_1 \leq 5$ bar: The flow remains supersonic over a part of the constant-area mixing chamber. This supersonic flow area grows with the primary pressure P_1 and ends by a pseudo-shock train which decelerates the flow. This pseudo-shock train reaches the entrance of the secondary diffuser for a stagnation pressure $P_1 = 5$ bar.
- For $P_1 > 5$ bar: The flow is choked over the whole length of the mixing tube and passes to subsonic through a quasi-normal shock at the beginning of the diffuser. In these conditions, the ejector operates with choking flow in the secondary nozzle. It may be noted that this limit value of P_1, corresponding to the occurrence of the flow regime with choking of the secondary nozzle, agrees quite well with the correlation suggested by Matsuo et al. (1981).

Results obtained in the case of free entrainment of induced air are presented in Fig. 4. In these Mach number fields' representations, we can visualize in the secondary nozzle convergent the secondary stream which is induced and which is flowing around the primary nozzle. This suction and entrainment of air flow causes a contraction of the primary jet at the primary nozzle exit. As for the ejector operating without induced flow ($U = 0$), it may be seen that the ejector operates in the subsonic regime for stagnation pressure $P_1 < 3$ bar. On the contrary, the mixing flow becomes supersonic in the constant-area chamber for values of stagnation pressure P_1 lower than for zero-secondary flow configuration. We can see the formation of a quasi-normal shock wave at the entrance of the secondary diffuser since for the stagnation pressure P_1 reaches 3 bar. Moreover, the flow pattern in the constant-area mixing chamber does not evolve significantly with increasing P_1, unlike for the zero-secondary flow configuration. On the other hand, increasing P_1 causes a significant increase in the shock wave strength and a noticeable shift of the shock wave in the downstream direction of the flow. In this case too, this shock appears as a quasi normal shock which causes the deceleration of the flow to subsonic velocity and then allows the mixture of primary and secondary fluids to achieve its recompression up to the atmospheric pressure along the diffuser.

Although this work is essentially a qualitative flow visualization study, some quantitative data about the flow and the ejector operation may be obtained from numerical simulations. Thus, when the ejector operates with no induced flow, the aspiration pressure P_2 is minimum (i.e. $P_2 = 0.17$ bar) for a primary stagnation pressure P_1 slightly inferior to 5 bar. This value of P_1 corresponds to the flow regime where the pseudo-shock train reaches the entrance of the secondary diffuser and also coincides with the maximum induced mass flow rate obtained in the case of free entrainment of secondary flow (i.e. $m_2 = 82.4$ kg/h, corresponding to an entrainment ratio $U = 0.46$).

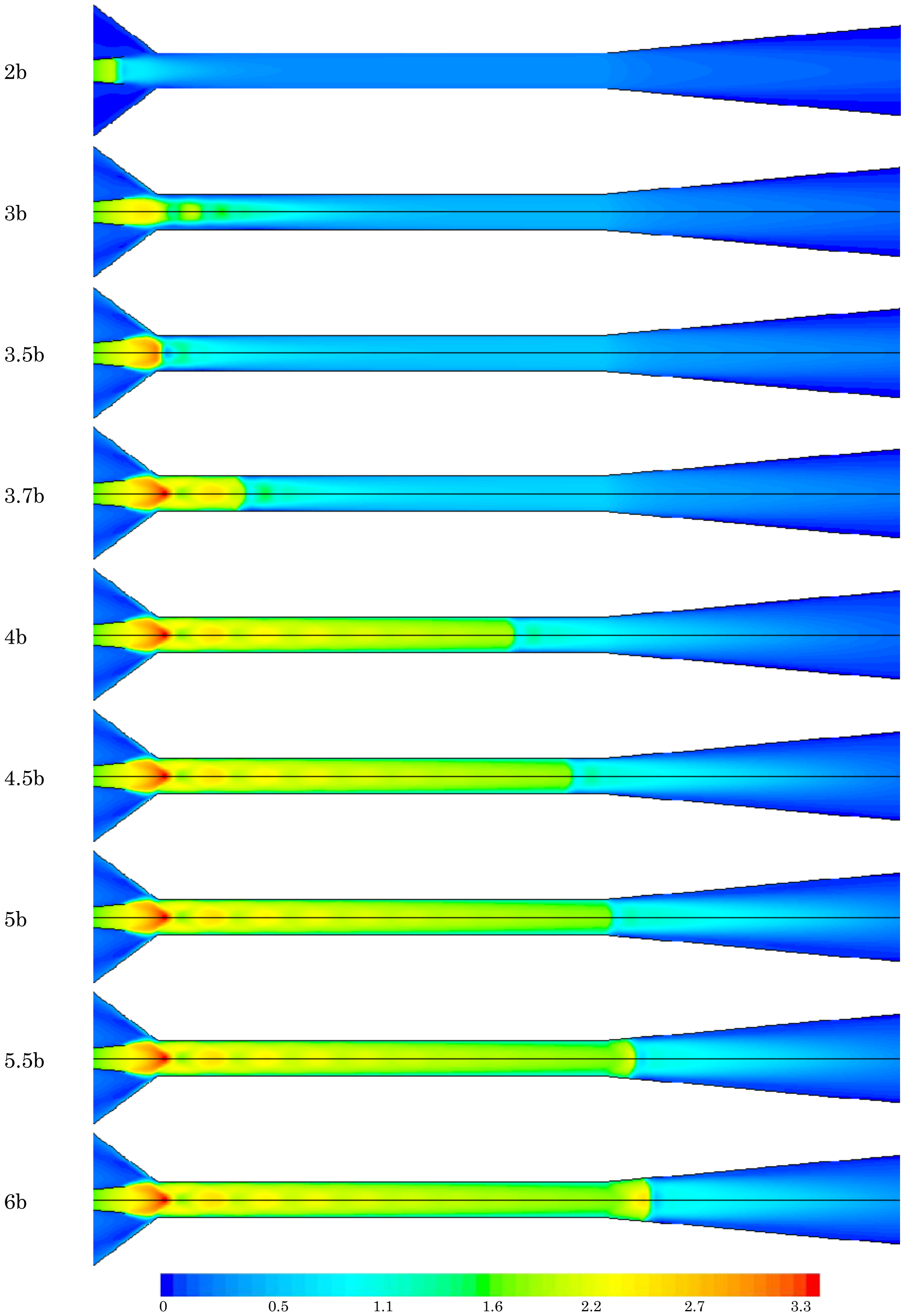

Fig. 3. Evolution of the Mach number field in the ejector with the primary stagnation pressure P_1 for zero-secondary flow configuration.

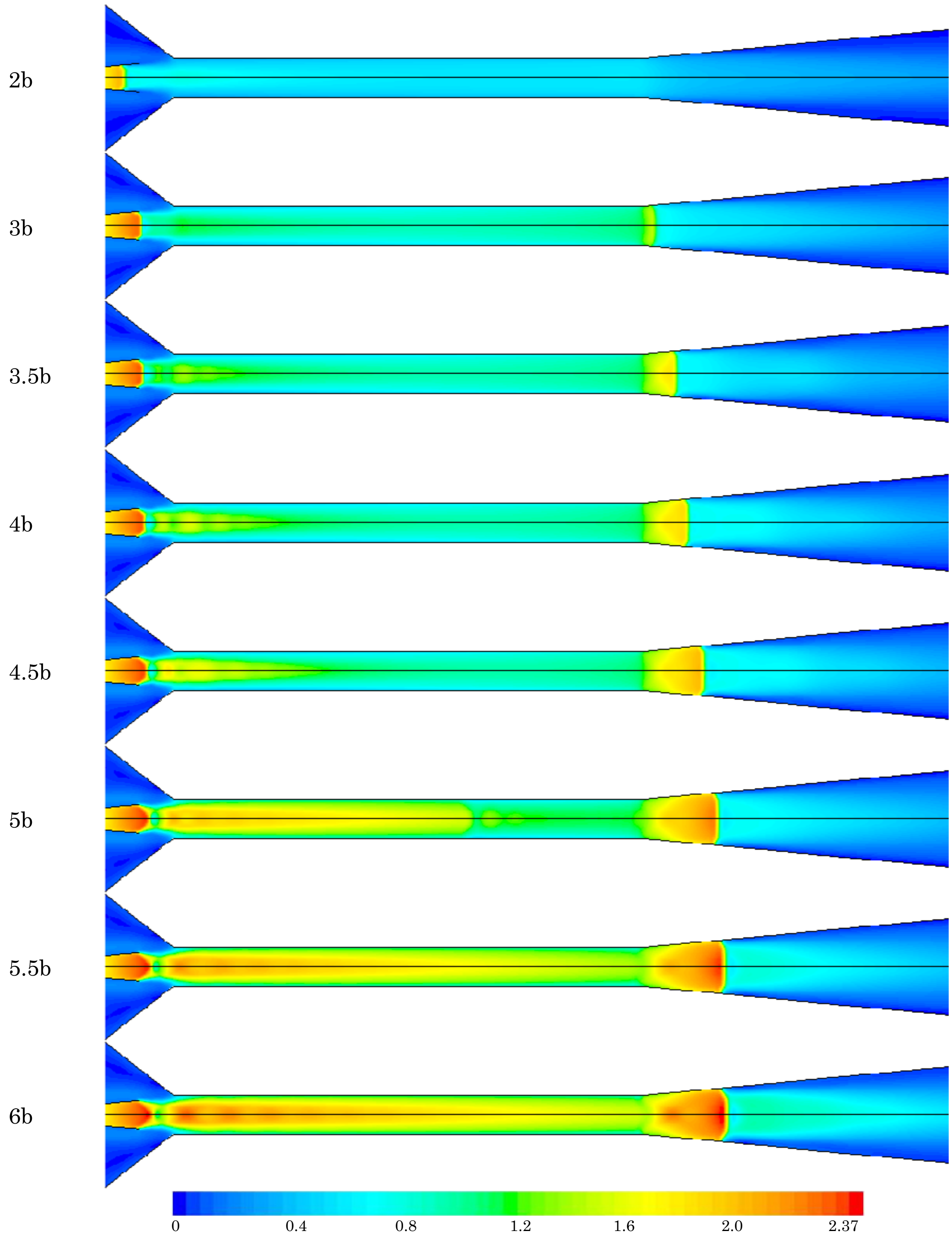

Fig. 4. Evolution of the Mach number field in the ejector with the primary stagnation pressure P_1 for free entrainment of secondary flow.

5. Conclusion

This paper deals with the numerical visualization study of the supersonic flow inside an air ejector. A CFD model of the air flow inside the ejector has been developed and validated by means of laser sheet flow visualizations. It has shown its capability to precisely predict the occurrence of the flow regime with choking of the ejector secondary nozzle. The computational results obtained for the prediction of the transition between flow regimes with and without choking are in good agreement with results found in the literature. In future work, we will apply this CFD model to investigate the effect of the mixing tube length on the flow structure and to examine the performance of supersonic ejectors in terms of suction and entrainment capacities for different operation modes.

References

Annamalai, K., Visvanathan, K., Sriramulu, V. and Bhaskaran, K. A., Evaluation of the performance of supersonic exhaust diffuser using scaled down models, Experimental Thermal and Fluid Science, 17 (1998), 217-229.

Bartosiewicz, Y., Aidoun, Z., Desevaux, P. and Mercadier, Y., CFD-experiments integration inthe evaluation of six turbulence models for supersonic ejectors modeling, Symposium on Integration CFD and Experiments, Glasgow, UK, (2003-9).

Chen, F., Liu, C.F. and Yang, J.Y., Supersonic flow in the second-throat ejector-diffuser system, Journal of Spacecraft and Rockets, 31 (1994), 123-129.

Chou, S. K., Yang, P. R. and Yap, C. R., Maximum mass flow ratio due to secondary flow choking in an ejector refrigeration system, International Journal of Refrigeration, 24 (2001), 486-499.

Desevaux, P., Formation de nano-gouttelettes d'eau au sein d'un éjecteur à air induit : une étude qualitative par visualisation de l'écoulement, Can. J. Chem. Eng., 79 (2001), 273-278.

Desevaux, P. and Aeschbacher, O., Numerical and experimental flow visualizations of the mixing process inside an induced air ejector, International Journal of Turbo and Jet Engines, 19 (2002), 71-78.

Desevaux, P., Lanzetta, F. and Bailly, Y., CFD modelling of shock train inside a supersonic ejector: Validation against flow visualizations and pressure measurements in the case of zero-secondary flow, Proceedings of the 10th International Symposium on Flow Visualization, Kyoto, (2002-8), F-0259.

Desevaux, P., Prenel, J.P. and Hostache, G., Flow visualization methods for investigating an induced flow ejector, Journal of Flow Visualization and Image Processing, 2 (1995), 61-74.

Eames, I.W., Aphornaratana, S. and Haider, H., A theroretical and experimental study of a small-scale steam jet refrigerateur, International Journal of Refrigeration, 18 (1995), 378-385.

Fabri, J., and Siestrunck, R., Supersonic air ejectors, In: Dryden HL, Von Karman T editors, Advances in Applied Mechanics, 5 (1995), 1-34, Academic Press, New York.

Matsuo, K., Miyazato, Y. and Kim, H-D., Shock train and pseudo-shock phenomena in internal gas flows, Progress in Aerospace Sciences, 35 (1999), 33-100.

Matsuo, K., Sasaguchi, K., Tasaki, K. and Mochizuki, H., Investigation of supersonic air ejectors – Part 1. Performance in the case of zero-secondary flow, Bulletin of JSME, 24 (1981), 2090-2096.

Matsuo, K., Sasaguchi, K., Kiyotoki, Y. and Mochizuki, H., Investigation of supersonic air ajectors – Part 2. Effects of throat-area-ratio on ejector performance, Bulletin of JSME, 25 (1982), 1898-1905.

Sankaran, S., Satyanarayana, Annamalai, K., Visvanathan, K. and Babu, V., Sundararajan, CFD analysis for simulated altitude testing of rocket motors, Canadian Aeronautics and Space Journal, 48 (2002), 153-162.

Sun, D.W. and Eames I.W., Recent developments in the design theories and applications of ejectors – a review, Journal of the Institute of Energy, 68 (1995), 65-79.

Author Profile

Philippe Desevaux: He is an Assistant Professor at the University of Franche-Comté. He received his Ph.D. degree in Engineering Sciences from the University of Franche-Comté in 1994. His research interests are in the field of fluid dynamics and include the development of new investigation techniques and their application to the solution of problems for aerodynamics, hydrodynamics and two-phase flows.

Alaoui Mellal: He received his M.S. degree in Mechanical Engineering in 2003 from the Scientific and Technical University of Rouen (INSA). He is now working on ejector applications and experimental investigation of supersonic flows by laser tomography visualization.

Yann Alves De Sousa: He received his M.S. degree in Mechanical, Energy and Environmental Engineering in 2003 from the University of Franche-Comté. His research interests are in the area of computational fluid dynamics, and more particularly concerning the modelling of internal supersonic flows and shock waves.